MicroStation for AutoCAD Users

Learn to be MORE Productive than EVER BEFORE!

Presented by

Jeanne Aarhus
ProSoft NET - Omaha

jaarhus@prosoftnet.com

ProSoft NET

SALT LAKE CITY ◆ PORTLAND ◆ SEATTLE ◆ BALTIMORE ◆ OMAHA

www.prosoftnet.com

MicroStation for AutoCAD Users

Preparing for a Successful Transition or "Co-Existence"

First and foremost……NO two CAD packages are exactly alike. But as technology progresses (and the "Windows" interface invades just about every application), MicroStation and AutoCAD are getting more and more similar with each new version. However, MicroStation and AutoCAD are still different CAD packages in some ways, specifically in design and usage. Your goal is to make this transition as seamless as possible. Learn the possibilities, the limitations, and most importantly learn the differences……and work with them.

For this presentation we will compare MicroStation V8 2004 and AutoCAD Release 2005. We will also discuss some aspects of "previous" versions where necessary.

In this session we will look at:

- Learning to "SPEAK" their language
- User Interface similarities
- Windows - Zooms
- Mouse Functions
- Keyins - Command Line
- AccuDraw - Direct Distance
- Shortcuts and Alias Commands
- Views - Viewports
- Seed Files – Templates
- Tentative Snap – Object Snap
- View Commands – Transparent Commands
- Handles – Grips
- Levels – Layers
- Line styles – Linetypes
- Enter Data Fields – Attributes
- LineWeight – Color
- Reference Files – Xrefs
- Cell Libraries - DesignCenter

What's NEW in V8 2004 for AutoCAD Users

Additional topics covered:

- DGN/DWG Enhancements
- Multiple Snaps
- DGN Compress – Purge
- Windows-Like Interface
- Text Editors
- *<Double-Click>* Editing
- Standards Checking
- ESC to Cancel commands

What Versions are "LIVE" in Today's CAD World

There are too many versions are out there! That is my first response too….but that's life right?

MicroStation	**AutoCAD**
95	Release 13
SE	Release 14
J or J 7.1.x	Release 2000 or 2000i
v8	Release 2002
v8.1	Release 2004
v8 2004	Release 2005

I know….and you know some Intergraph/Bentley users out there may still be using IGDS or MicroStation v4.0! And…likewise, there are still some AutoCAD users out there in Release 10….but we have to draw the line somewhere! So we will limit this discussion somewhat ok?

What Applications are "LIVE" in Today's CAD World

MicroStation	**AutoCAD**
Triforma	Architectural Desktop
GEOPAK	Land Development Desktop
INROADS	INROADS
	Revit

Learn to Speak "THEIR" Language

One of the first barriers for MicroStation and AutoCAD users is the language barrier. The table below shows how a simple conversation can result in a multitude of misunderstandings on BOTH sides. Learning to speak the "other" language can minimize these misunderstandings and simplify the whole process.

MicroStation	AutoCAD
Elements	Objects or Entities
Levels	Layers
Attributes	Properties
Cells	Blocks or WBlocks
Active Design File	Drawing File or Drawing Database
Reference Files	Xrefs or Reference Files
Seed File	Prototype or Template File
Drop	Explode
Active	Current
Tags	Attributes
Fit	Zoom Extents
Window Area	Zoom Window
Fence	N/A – closet match is the Selection Window/Crossing
Pattern	Hatch
Solid Fill	2D Solid or Solid Hatch
Selection Handles	Grips
Fence or Selection INSIDE or Polygon Selection (SE/J PowerSelector)	Selection Window
Line Selection (SE/J PowerSelector)	Selection Fence
Fence or Selection OVERLAP or Polygon Selection (SE/J PowerSelector)	Selection Crossing
Fence or Selection CLIP VOID VOID OVERLAP VOID CLIP	N/A EXCLUDE Window (Express Tools only) EXCLUDE Crossing (Express Tools only) N/A
BYLEVEL or BYCELL	BYLAYER or BYBLOCK
DEFAULT level	LEVEL 0 (zero)
ModelSpace/PaperSpace	Models
Layout	Sheet

How do "Graphics" Compare DGN → DWG?

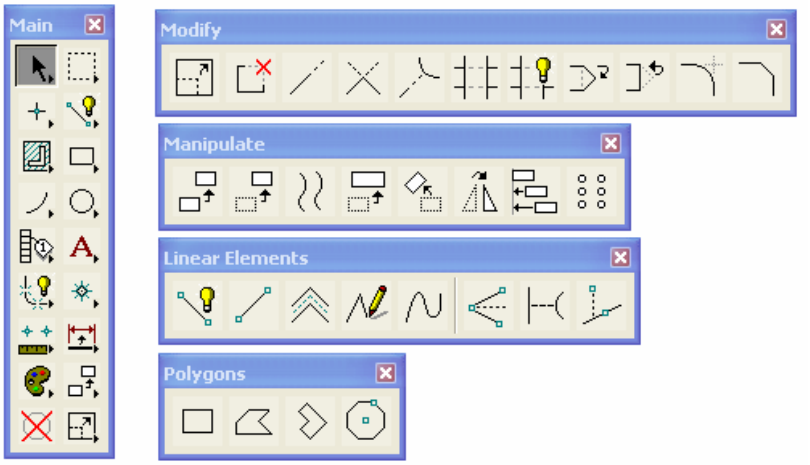
This table displays the comparable elements to objects found in MicroStation and AutoCAD.

MicroStation - DGN	AutoCAD - DWG
Line	Line
Line (Zero Length) (DL=0), Point	Point
Open Smartline Line String	Polyline or LWPolyline
Closed Smartline - Shape / Complex Chain	Polyline or LWPolyline1
Text	Text
Text Nodes	MText
Enter Data Fields	N/A (closet match is Attributes)
Shape (< 4 vertices)	Polyline, Face, Polyline Mesh, Solid
Shape (> 4 vertices)	Polyline, Polyline Mesh
Shape / Hatch	Polyline / Lines, Hatch
Shape / Xhatch	Polyline / Lines, Hatch
Shape / Pattern	Polyline / Hatch
Shape / Opaque Fill	Polyline / Solid Fill Hatch
Cell	Block or WBlock
Shared Cell	Block or WBlock
Cell Library	N/A or DesignCenter - Single block stored per file
Circle	Circle
Ellipse	Ellipse (> R13), Polyline (< R13)
Arc	Arc
Curve	Spline
Curve Stream	Spline
Dimension	Dimension
Ellipse	Ellipse (> R13), Polyline (< R13)
Multi-line	Polyline(s) or Multi-line
Tag	Attribute or Text
Ellipse	Ellipse (> R13), Polyline (< R13)
Reference File (design file)	Xref
Self-Referenced File	N/A
Reference File (sheet file)	Viewport
B-Spline	Spline
N/A	Fields
N/A	Tables

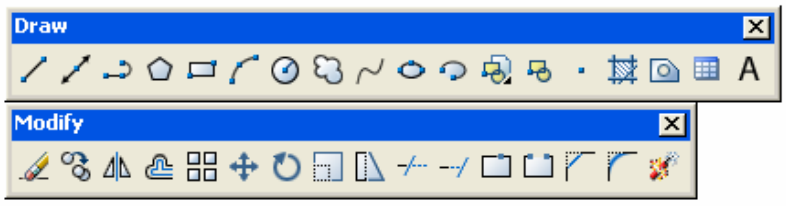
TOOL BOXES vs. TOOLBARs

Frankly, these aren't that different…isn't everything "Windows-Like" anymore? Let's take a look.

MICROSTATION: Draw and Modify Toolboxes



AUTOCAD: Draw and Modify Toolbars



See…they aren't that
different right? You just have to find the commands in this new environment.

Your NEW User Interface

Two CLASSES of Commands

MICROSTATION: In MicroStation, not ALL commands are alike. There are basically two classes of commands; PRIMARY and VIEW commands.

- **PRIMARY** commands include those that draw, modify and "manipulate data" in the design file.
- **VIEW** commands include those that zoom, pan and "move around" in the design file.

How does this affect your work? Basically, you should consider all VIEW commands as "transparent" commands; which means they will run inside of PRIMARY commands without stopping the PRIMARY command. Once the VIEW command is complete, the PRIMARY command will pick up where you left off and continue.

AUTOCAD: In AutoCAD, ALL commands are pretty much alike. Yes, some can be run "transparently", but very few.

ALL ZOOMs are NOT ALIKE!

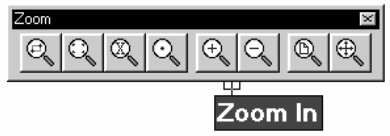
The functionality of the view manipulation commands are very similar. The differences focus primarily on "how many views" you get and how many clicks it takes to execute.

MICROSTATION: Let's look at ZOOM IN and ZOOM OUT. MicroStation will ZOOM IN or OUT about the *<Left-Click>*

AUTOCAD: ZOOM does not work the same in AutoCAD. First, the ZOOM IN and ZOOM OUT buttons always zoom about the CENTER of the view…you don't get a chance to *<Left-Click>* and zoom about a specific point.

Note: Do you know how to make the AutoCAD ZOOM work just like a MicroStation ZOOM? Yep!!! Modify the button in AutoCAD to run the following command

ZOOM CENTER .5 ; this zooms in .5x about your <Left-Click>



There are a lot of situations just like this one! I will try to point them out as we go along!

DYNAMIC Zooming

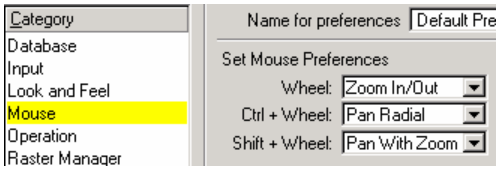
Both packages offer some wonderful dynamic zoom options

MICROSTATION: Dynamic panning can be accomplished using two methods:

First, hold down the <*SHIFT*> + <*Right-Click*> and DRAG your cursor across the screen. You should immediately see the screen begin to pan. To control the direction of the pan using this method, think of it as "driving down the street" using your cursor as the car.

Second, hold down the <*CTRL*> + <*Wheel*> on the mouse. Rolling the wheel UP will pan UP; and rolling the wheel DOWN will pan DOWN.

Check **Workspace → Preferences → Mouse** to modify any of the default "wheel" functions.



Exercise: Controlling your VIEW

Let's try a simple exercise and "tweak" the default settings.

STEP 1: **File → Open**….**ZOOM.DGN**

STEP 2: Place the cursor over the "rectangle" and roll the mouse wheel to **ZOOM IN** and **ZOOM OUT**

You should notice that the default function of the wheel performs a PAN function rather than a ZOOM function. To change this default you must modify your user preferences.

STEP 3: Select **Workspace → Preferences**

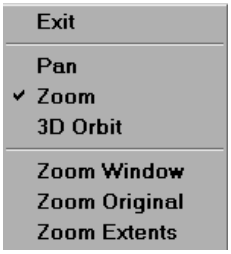
STEP 4: Select the **MOUSE** category, and locate the settings for **Mouse Preferences**

STEP 5: Modify the **WHEEL** setting to **ZOOM IN/OUT**

STEP 6: Pick **OK** and answer **YES** to save the changes

AUTOCAD: AutoCAD also offers some GREAT Real-time ZOOM and PAN commands.

ZOOM REALTIME and PAN REALTIME….



ZOOM works slightly different in any AutoCAD 200x. These releases of AutoCAD introduced the "wheel" mouse functionality. When using ZOOM, you can place the cursor over the "area" you want to ZOOM on and roll the wheel forward for ZOOM IN and backward for ZOOM OUT. This allows you to zoom in on anything….not just the CENTER of the view….FINALLY!

Pressing the "wheel" like a button will also initiate the PAN command.

While using the <CTRL> key + <Wheel> will initiate DYNAMIC PANNING similar to MicroStation.

MICROSTATION J: An MDLAPP is available for J that enables the "wheel" functions for MicroStation similar to V8. This will only work however if you are using Windows 98, Windows NT, or Windows 2000.

Exercise: Improving your ZOOM in MicroStation J

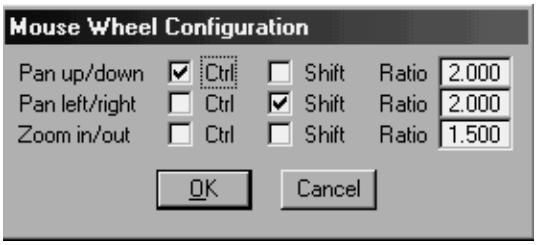
This exercise will show you how to get your wheel mouse working in J…just like in V8!

STEP 1: Download the following application from the select web site if needed, and try it out!

IMOUSE.MA

STEP 2: Keyin **MDL LOAD IMOUSE**

STEP 3: Keyin **IMOUSE CONFIG**



PERSISTANT vs. <Right-Click>

Persistent Commands are what cause MicroStation to remember the last command you were in. For example, if you pick the PLACE LINE command you will remain in the PLACE LINE command until you pick another command. AutoCAD doesn't support "persistent" commands. This can be one of the most frustrating differences when switching back and forth between the two applications. Here are some suggestions.

MICROSTATION: You don't need to modify anything to get this to work. Persistent commands are the DEFAULT. If you really want to you can turn this off. Here's how:

Exercise: Controlling the *<Right-Click>* and *BUTTON* feel in MicroStation

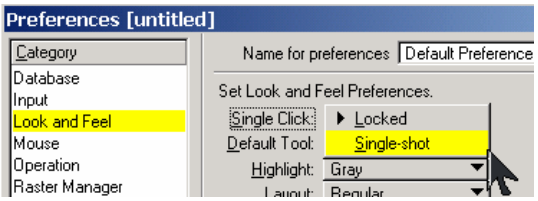
This exercise will demonstrate how you can "tweak" your mouse and tool LOOK and FEEL.

MICROSTATION:

STEP 1: Select **Workspaces → Preferences**

STEP 2: Select the **LOOK and FEEL** category, and locate the settings for **Single Click**

STEP 3: Modify the **Single-Click** option to **Single-Shot**



Now MicroStation will NOT have "persistent commands", and it works just like AutoCAD.

Note: I don't personally use it this way since I think the default is more efficient; but try it out and set your personal preference.

AUTOCAD: There are two ways to "simulate" persistent commands in AutoCAD.

First, *<Right-Click>* anywhere on the screen and select the REPEAT COMMAND option. OR….hit the *<Enter>* key (most AutoCAD users also use the *<Space-Bar>*) for this one since the *<Space-Bar>* in AutoCAD is equal to the *<Enter>* key. And after all…your left hand isn't doing anything else anyway right?

Second, place an "*" in front of a command (on a button?) and the command will repeat itself.

For example: I will modify the RECTANGLE command

I will modify the button to read:

*^C^C_RECTANG

Not all commands will allow you to do this. But it makes an old MicroStation user much happier in "AutoCAD".

All UNDOs are NOT ALIKE

Be careful on this one!

MICROSTATION: The UNDO command in MicroStation will only UNDO the PRIMARY commands not the VIEW commands.

To UNDO a VIEW command you need to use the VIEW UNDO command located in the scroll bar buttons.

AUTOCAD: The UNDO in AutoCAD recognizes ALL COMMANDS; including UNDO, ZOOM, PAN, etc.

All REDOs are NOT ALIKE

Be VERY careful on this one!

MICROSTATION: The REDO command in MicroStation will REDO as many times as you like.

AUTOCAD: **Release <2004:**
The REDO command in AutoCAD will only REDO the last UNDO. That is it! You get ONE and ONLY ONE…REDO!

So be careful to not UNDO to quickly or you are sunk!

Release 2004 - 2005 :
The newest releases of AutoCAD now supports unlimited REDO commands exactly like those found in MicroStation.

Exercise: Understanding UNDO and REDO

This exercise will help you understand how MicroStation uses the UNDO function; which is different than AutoCAD.

MICROSTATION:

STEP 1: **File → Open….UNDO.DGN**

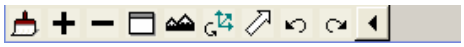
STEP 2: Take a few minutes and add several elements to the drawing
Then delete and move a few

STEP 3: Now ZOOM and PAN around the view

Select **Edit → Undo**…or pick the **UNDO** button. 

You should notice that ZOOM and VIEW commands are ignored by the MicroStation UNDO command. This is not a mistake…it is intentional. Once you get used to it you will like this better….I promise!

There is a separate UNDO for ZOOM and VIEW commands, and it is located on the scrollbar portion of each VIEW. 



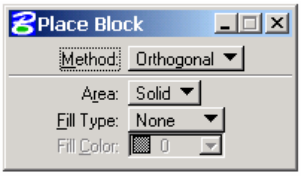
STEP 4: You can also use an alternative PAN command which is identical to the one you used to have in AutoCAD. It is also located on the scrollbar portion of each VIEW.

Note: If you are interested in another shortcut….try using <CTRL-Z> to perform the UNDO. Much quicker right?

KEYIN vs. COMMAND LINE

Both applications allow for most commands to be keyed in by the user.

MICROSTATION: MicroStation has made an incredible effort in allowing the user to choose their method of input; keyins, buttons, pulldowns, etc. Many users have chosen to move away from this keyin environment…and therefore, be more Windows-like.



AUTOCAD: AutoCAD still relies quite heavily on the command line or "keyin" interface; but they are also trying to move away from the keyin user interface.

As you can see both applications use toolbars and buttons. However, AutoCAD doesn't provide a Tool Settings Utility that allows the user to completely avoid keyin commands.

The later versions of AutoCAD 200x did implement a robust series of *<Right-Click>* Menus which allows the user to avoid most keyins.

Many of the new commands and interface changes have been in the introduction of *<Right-Click>* menus or Pop-Up menus. The command line is still a relevant and necessary part of AutoCAD, but the change is significant.

DL= vs. @X,Y

This section includes the equivalent keyin commands previously used in both MicroStation and AutoCAD. To avoid these keyins…see the next section….but sometimes you have to use these…so here they are!

A significant difference occurs here due to the Working Unit vs. Unit files.

MICROSTATION: When I enter the value of "1" unit length in MicroStation, that is actually 1 MASTER UNIT.

"1" unit to a Civil Engineer in AutoCAD may be 1 foot, while
"1" unit to a Mechanical Engineer in AutoCAD may be 1 inch; and the files are setup almost identically. You need to know this from the AutoCAD user.

However; in MicroStation "1" unit is typically 1 ft….and ":1" unit is 1 in.

WATCH OUT! This is the most common cause for the "factor of 12" errors we get when using alternate applications files.

Absolute Coordinates	**XY=X,Y**
Distance DELTA Length:	**DL=X,Y**
Distance Direction:	**DI=distance,direction**

Note: You can use the ";" in place of the ":" key to speed up precision entry.

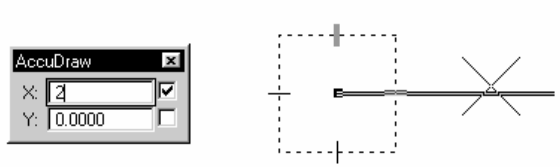
AUTOCAD: As you can see below, the keyins are similar in functionality, but different in syntax:

Absolute Coordinates	**X,Y**
Relative Coordinates	**@X,Y**
Polar Coordinates	**@length<angle**

ACCUDRAW vs. DIRECT DISTANCE

In the latest releases of both applications, some significant user productivity tools have emerged.

MICROSTATION: AccuDraw allows the user to draw precision elements without all the unnecessary keyins. This is one of the MOST powerful aspects of MicroStation…..it can save you HOURS in every day productivity.

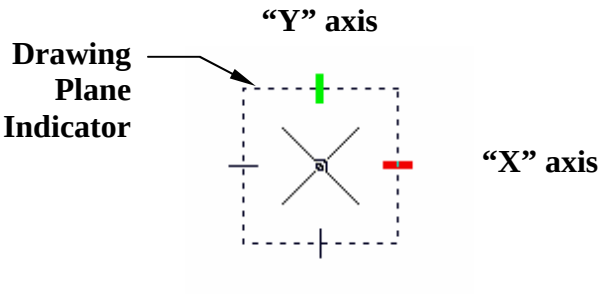


Some of the AccuDraw shortcuts that are a must for an AutoCAD user:

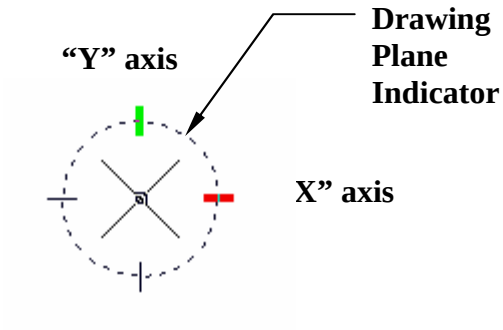
- O – defines a "temporary" ORIGIN similar to acquiring an OSNAP in AutoCAD
- V, T – rotates the compass back to the VIEW or TOP orientation

Exercise: Switching between POLAR and RECTANGULAR modes

When moving in the positive or negative direction, AccuDraw tracks your movements and automatically assigns a positive or negative value to the keyin you provide. Pretty neat, huh? But if you're like me, you better start paying attention to what you do with your cursor in between commands. Don't worry; it just takes a little getting used to!



Rectangular Compass



Polar Compass

STEP 1: Use the <*Space-Bar*> to toggle between **RECTANGULAR** and **POLAR** modes.

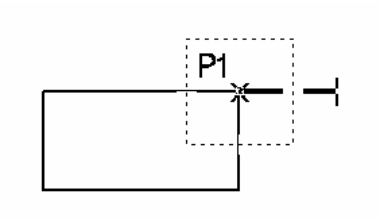
Note: My best tip for working with AccuDraw is TO IGNORE IT and it works better! The more you try to pick in the AccuDraw toolbar, the more you actually interfere with it. Learn the shortcut keyins and try to avoid "touching" it.

Exercise: Using the ORIGIN shortcut…

Use the **O** shortcut to locate "reference points" while using the AccuDraw direct distance keyins discussed earlier.

STEP 1: **File → Open**….**ORIGIN.DGN**

STEP 2: Select the **LINE** command and **TENTATIVE SNAP** to **P1** but **DO NOT ACCEPT IT…**



STEP 3: Keyin **O** to set the temporary reference origin at this point

STEP 4: Drag the cursor in the → X-direction and keyin a distance of **1**

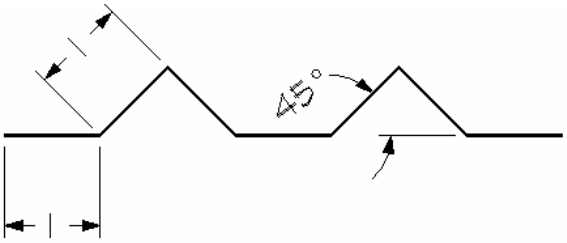
STEP 5: ***<Data-point>*** to start the line at this location

If your thinking of practical applications….remember this little tip!

Whenever you find yourself drawing a "temporary construction line" just to get a location to start drawing something else…….USE THIS INSTEAD!!!!!

Exercise: Re-Aligning AccuDraw…

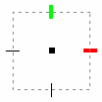
Let's try to draw this example:



STEP 1: **File → Open**….**POLAR.DGN**

STEP 2: Select the **LINE** command

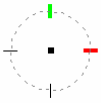
STEP 3: Hit the <*Space-Bar*> until you are in **RECTANGULAR** mode in AccuDraw



STEP 4: Start the line anywhere in your drawing

STEP 5: Drag the cursor in the → X-direction and keyin **1** for the distance

STEP 6: Hit the <*Space-Bar*> and switch to **POLAR** mode in AccuDraw

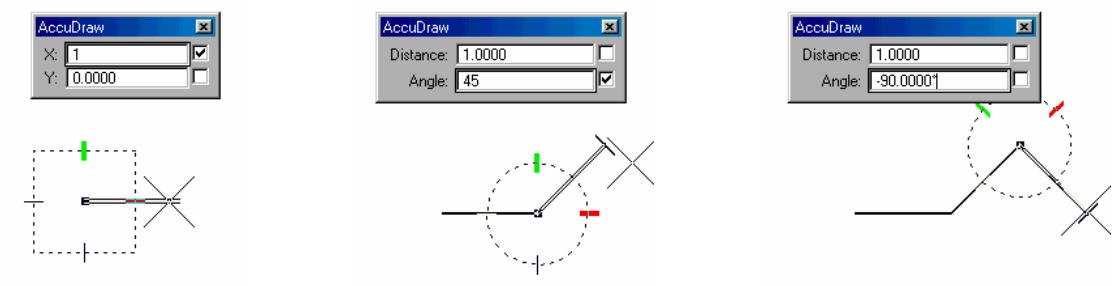


STEP 7: Hit the <*TAB*> key to move the focus in AccuDraw to the ANGLE field if necessary

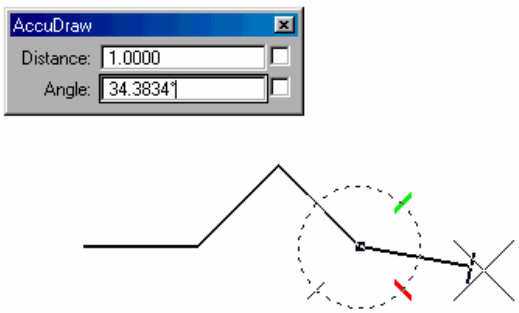
STEP 8: Keyin **45** to lock the angle to 45 degrees

STEP 9: Use the **REPEAT DISTANCE INDICATOR** to repeat the distance of **1**

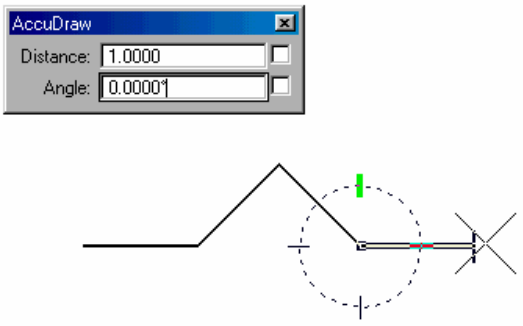
STEP 10: Again…use the REPEAT DISTANCE INDICATOR to repeat the distance of 1 for the third line segment



STEP 11: We need to re-align the AccuDraw compass with the **VIEW** to draw the next segment

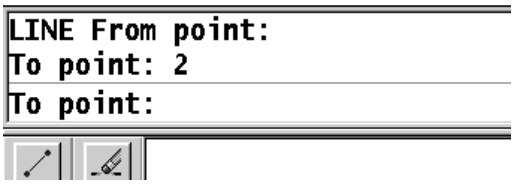


STEP 12: Keyin **V** to align the compass with the **VIEW**
….isn't this thing great!



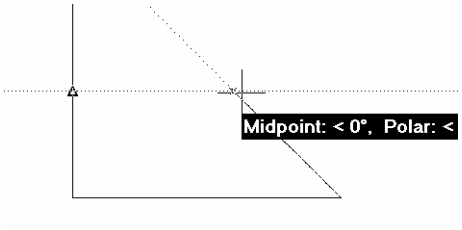
STEP 13: Finish drawing the rest of this exercise

AUTOCAD: Direct Distance allows the AutoCAD user to draw precise objects without having to use those "awful" @ keyins! This tool is typically used with ORTHO ON to give it full benefit! ORTHO is the equivalent to the MicroStation AXIS LOCK set to 90.

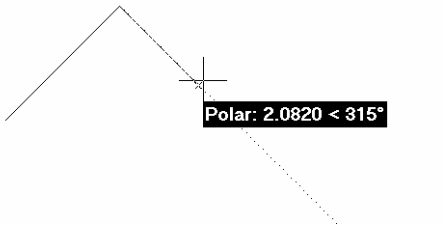


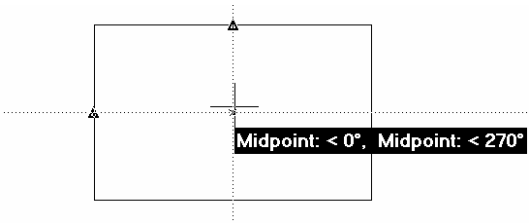
Direct Distance has been taken to a new level in 200x with the addition of POLAR TRACKING and OBJECT TRACKING. With these new tools AutoCAD has almost eliminated the need for construction objects in AutoCAD.

Here's how OBJECT TRACKING works….



Here's how POLAR TRACKING works….





You can even use both options together!

SHORTCUTs vs. ALIAS COMMANDS

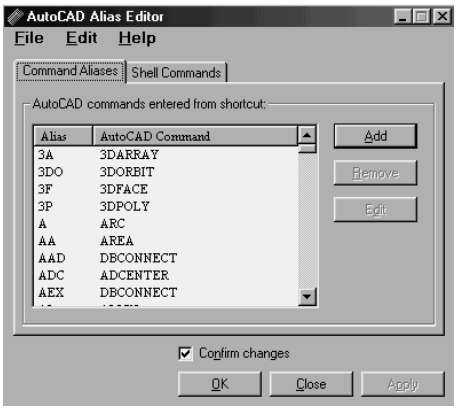
Both applications give you "shortcut" keyins, or "alias" commands to make your CAD life easier.

MICROSTATION: To access the AutoCAD keyins in MicroStation you can use the AutoCAD workspace to help a little. This workspace changes the tooltips to "AutoCAD" friendly tips.

Most MicroStation commands are 3-character specific…so just keyin the first 3 letters of the command and you can skip the rest. Some are single-character specific…but just a few!

AUTOCAD: Likewise, you can do the same in AutoCAD. AutoCAD provides an alias file called ACAD.PGP which lists all the alias commands for AutoCAD. If you are a MicroStation user, just edit this file as needed. Sorry, they don't provide one for you!

```
R,      *Redraw
Ra,     *Redrawall
Re,     *Regen
Rea,    *Regenall
Rec,    *Rectangle
Reg,    *Region
```



Note: There is a dialog box interface available to simplify the editing of this file. This utility is available with the EXPRESS TOOLS; and is available free of charge to subscribers.

Using ACAD.PGP

MICROSTATION: You can use your ACAD.PGP file in MicroStation by setting the following configuration variable.

MS_DWG_PGPFILE=c:\Program Files\AutoCAD 2002\support\acad.pgp

The next configuration variable that you might want to change is the "command prefix" setting for using the keyins from the ACAD.PGP file. The default is set to "\". You can change this to anything convenient.

MS_DWG_COMMANDPREFIX="\"

For Example: You want to be able to keyin E for ERASE?
Just keyin \E and the MicroStation DELETE ELEMENT is executed

Another Example: You can also use the built in keyins DWG ERASE, DWG LINE…etc. See the online help for a complete list of all DWG commands.

Note: You can also use this feature to create shortcuts for MicroStation commands.

MICROSTATION:

Exercise: Using SHORTCUT ALIAS in MicroStation

This exercise will help you set up your MicroStation to use the AutoCAD commands you already know. But remember, you should still try to convert your language over to the MicroStation terminology so you begin the transition…sooner rather than later.

STEP 1: **File → Open**….**KEYINS.DGN**

STEP 2: Select **Workspace → Configuration**

STEP 3: Select the category **DWG/DXF** and highlight the setting for **PGP Command Alias File**

STEP 4: To modify this setting pick the **SELECT** button and navigate to the training file listed below:

C:\BIUC\uSTN_for_ACAD_Users\ACAD.PGP

STEP 5: Pick **OK** and **YES** to save the changes

STEP 6: Keyin **\E**….this should execute the **DELETE ELEMENT** command

STEP 7: Try another one, Keyin **\X**….this should execute the **DROP ELEMENT** command

Pretty cool…right?

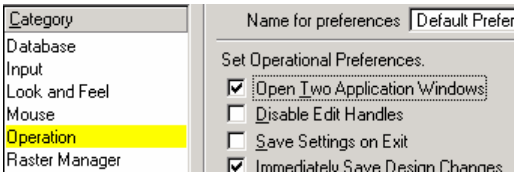
Note: Don't forget you can also use the prefix DWG

VIEW vs. VIEWPORT

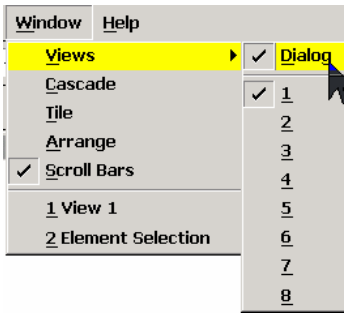
MICROSTATION: Allows for multiple Windows, and Dual-Monitors. It is possible to draw between views and monitors quite easily. The focus change is automatic and no extra clicks are required. There is a user preference setting to allow for dual application windows.

Workspace → Preferences → Operation

MicroStation even works with two application windows on a Single-Monitor system…this is great way to get all those dialog boxes out of your way, and off your drawing area!

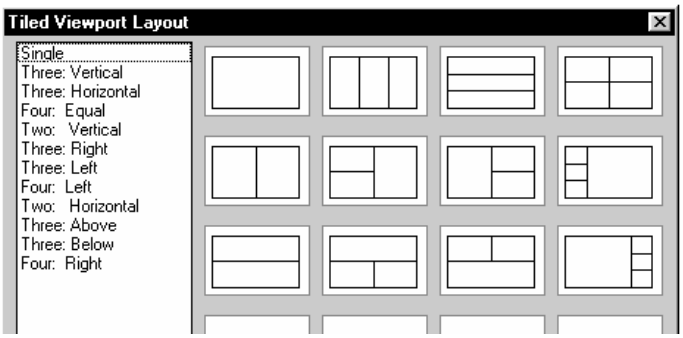


Another nice feature is the multiple windows available in MicroStation. Try this out…you will love it!
Window → Views → Dialog



AUTOCAD: You can have multiple viewports in AutoCAD but the focus is allowed in only "1" viewport at a time. To draw from one viewport to another the user must *<Left-Click>* in the viewport to change focus, and then *<Left-Click>* again to continue a command. Similar, but a little cumbersome, and a lot more clicks!

Most AutoCAD users don't use multiple viewports in modelspace. However, there are several predefined viewport arrangements you can choose from.



View → Tiled Viewports → Layout

You can also have dual-monitors for AutoCAD, but this is related more to the operating system and video card combinations. Not typical in the AutoCAD world…but gaining in popularity!

SEED File vs. TEMPLATE/PROTOTYPE

Similar procedures take place here…both systems use "BASE" files to create NEW files.

MICROSTATION: SEED files store the desired settings for "future" design files. SEED files have a .DGN extension by default.

AUTOCAD: TEMPLATE or PROTOTYPE files store the settings for "future" design files. These files can be inserted into "older" files to force or inherit "template" file settings. Not all settings can be inherited this way. TEMPLATES have the file extension .DWT; while PROTOTYPES have the file extensions .DWG.

TENTATIVE SNAP vs. OBJECT SNAP

Achieving exact precision requires the user to "snap" to all elements while drawing.

MICROSTATION: MicroStation has two methods for snapping to elements.

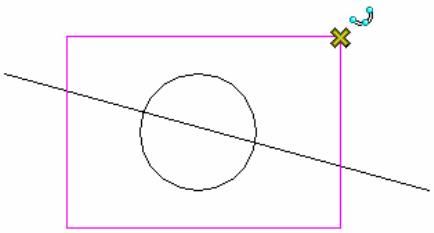
First, the TENTATIVE SNAP uses a combination of mouse buttons to highlight and snap to elements. MicroStation allows you set ONE snap "running constant" and have a SECOND snap "single-shot".

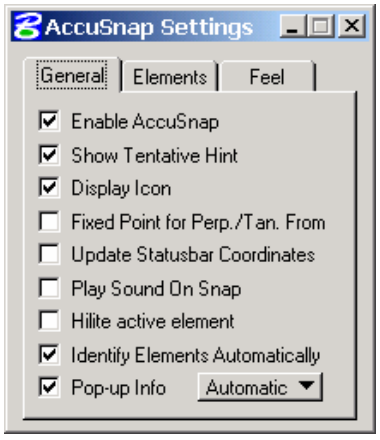
"EXTRA" Snaps Available in MicroStation:

Keypoint	(multiple keypoints…equivalent to ACAD center;endpoint;midpoint;and quadrant)
Perpendicular FROM	
Tangent FROM	
Bisector	
Through Point	(similar to OBJECT TRACKING in 2000)
Point On	(similar to OBJECT TRACKING in 2000)

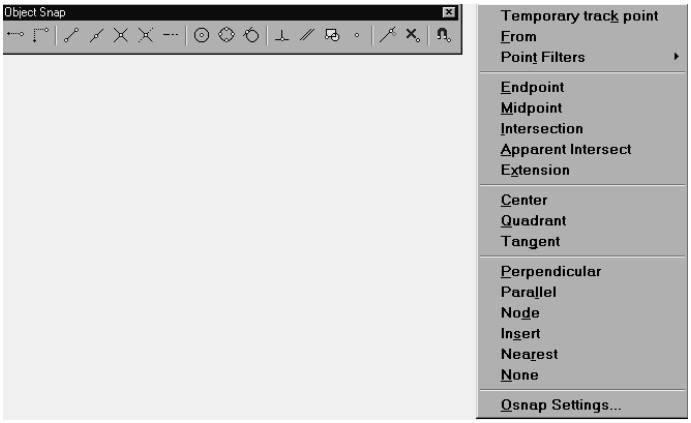
Second, the new ACCUSNAP works very similar to what AutoCAD users are used to. This tool will display SNAP ICON representing the active snap setting. You have several ACCUSNAP settings to allow you to "tweak" how this new tool works.

It also displays additional information about the element; including type [level]





AUTOCAD: Object Snaps are AutoCAD's way of "snapping" to objects. AutoCAD allows you to have as many Object Snaps set RUNNING as you like. You are not limited to "two" like in MicroStation.



Object Snaps in 200x saw the biggest improvement in years. The addition of PARALLEL and EXTENSION added some missing functionality.

Extra Object Snaps available in AutoCAD 200x:

AccuDraw and AccuSnap provide a "close" match for these -

Temporary Track Point
From
Point Filters
Osnap Markers
Extension
Node (similar to a POINT in MicroStation)

Exercise: Getting TENTATIVE to work…

This exercise will help you get used to the need for the TENTATIVE SNAP, and how you can change how it functions if you just CAN'T get your fingers to do both buttons at once!

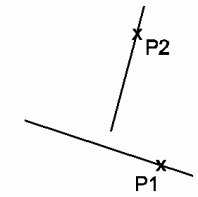
STEP 1: **File → Open**….**TENTATIVE.DGN**

STEP 2: Select **Workspace → Button Assignments**

STEP 3: Highlight **TENTATIVE** from the list of buttons and move your cursor over the **BUTTON DEFINITION AREA**

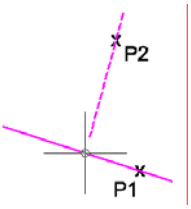
STEP 4: Press the mouse button that you prefer to use for your **TENTATIVE** button

STEP 5: Pick **OK** to save the changes



Now let's try to use the TENTATIVE SNAP to snap to the intersection of non-intersecting lines.

STEP 6: **TENTATIVE SNAP** to **P1** and **TENTATIVE SNAP** to **P2**.
If you performed this maneuver successfully the lines should appear dashed and/or highlighted



VIEW COMMANDs vs. 'TRANSPARENT COMMANDs

MICROSTATION: When running a PRIMARY command (draw or manipulation command). MicroStation allows you to use SECONDARY or VIEW commands (view or zoom) without stopping the PRIMARY command.

AUTOCAD: When running any AutoCAD command, there is no such thing as "primary" or "secondary" commands. However, AutoCAD does allow you to run several commands in "TRANSPARENT" mode. This allows the AutoCAD user to run a TRANSPARENT ZOOM while in a LINE command. To execute any command in "TRANSPARENT" mode just place a ' before the command…it will be TRANSPARENT.

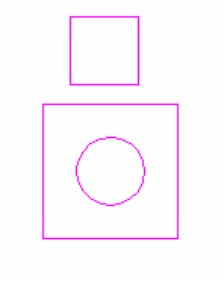
'ZOOM

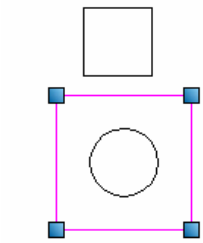
Note: Some commands can not be run in transparent mode
AutoCAD will warn you if you try one that is not allowed!

HANDLEs vs. GRIPs

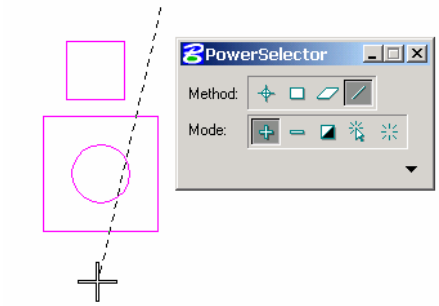
Most users pick the command first, then pick the element or object they want to manipulate. However, both applications allow the "other direction" as well.

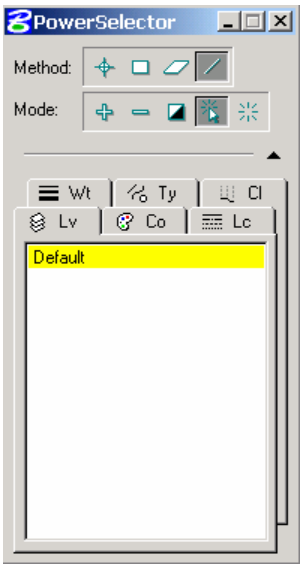
MICROSTATION: Use the SELECTION TOOL in MicroStation…I recommend using the POWERSELECTOR in SE/J/V8…it's great! Most AutoCAD users will appreciate this tool since it performs some of the same selection methods they are used to.

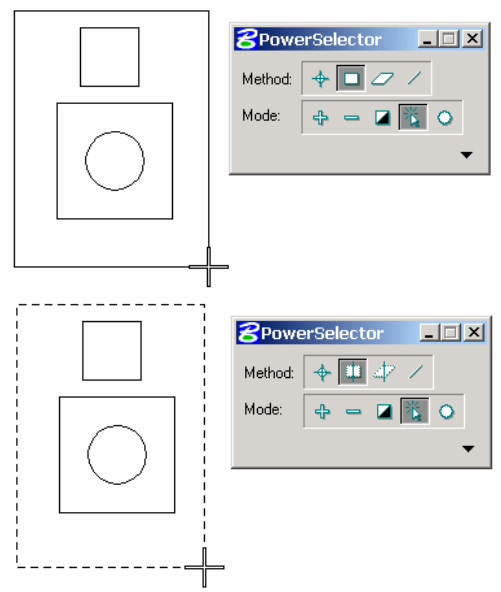


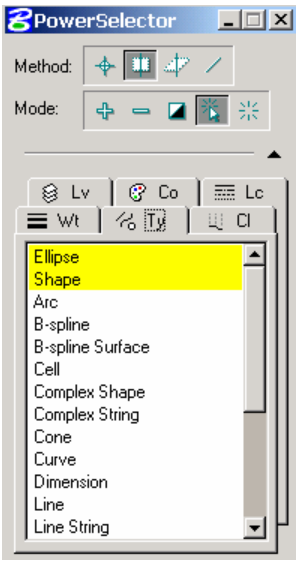


Here are
some samples of what you can do
with POWERSELECTOR

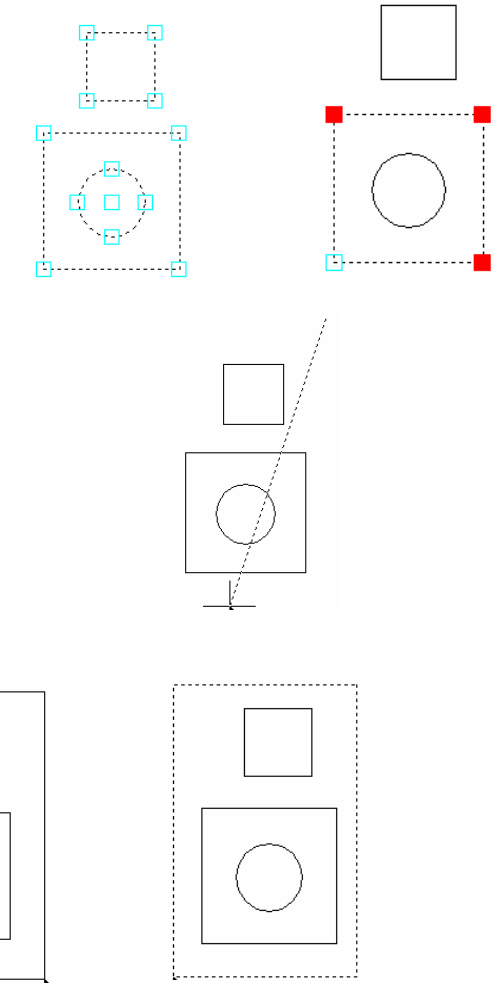


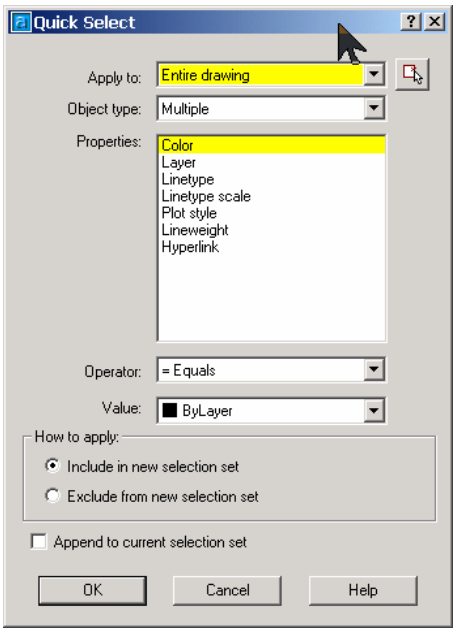






AUTOCAD: Using GRIPS in AutoCAD is really easy….I'm still a CAD user from the old school…so I still pick the command first, and then the object when possible. But, with GRIPS, there are some things I can do that the "normal" AutoCAD way won't let me do!





Exercise: Using POWERSELECTOR

This exercise will focus on how to use the POWERSELECT tool.

STEP 1: **File → Open**….**POWERSELECT.DGN**

STEP 2: Select the **POWERSELECTOR** tool, it may be hidden UNDER the SELECTION tool 

STEP 3: Take a look at the Tool Settings dialog. There are several options here that should remind you of AutoCAD…and more!

Let's test your selection skills!

STEP 1: Select **ONLY** the **circles**….can you do it? What option did you use?

STEP 2: Select **ONLY** the **lines**….can you do it? What option did you use?

STEP 3: Select **ONLY** the **hexagons**….is it getting easier? Hope so!

STEP 4: Take a few minutes to try it on your own….

You can change the default SELECTION tool to **POWERSELECTOR** if you liked it better…here's how:

STEP 1: Select **Workspace → Preferences**

STEP 2: Select the category **LOOK and FEEL** and locate the **DEFAULT TOOL**

STEP 3: Modify the **DEFAULT TOOL** to **POWERSELECTOR**

STEP 4: Pick **OK** to save the changes

Basics of LEVEL vs. LAYER

One of the most fundamental aspects of any CAD package is the “layering” scheme used to control the display of graphics.

V8 has an “unlimited” number of levels you can use. You are no longer limited to 63 levels per file. Don’t go to crazy with this, but more than 63 levels is a significant enhancement. Levels can now work similar to AutoCAD 200x. You can use the BYLEVEL or BYCELL level settings to enable you to easily exchange data with AutoCAD users. Try it out…this works really well!

Level Names can be 1-511 characters in length, and you can group several levels together using Level Filters, or Level Filter Groups.

To change the ACTIVE LEVEL use the PRIMARY TOOL BOX:

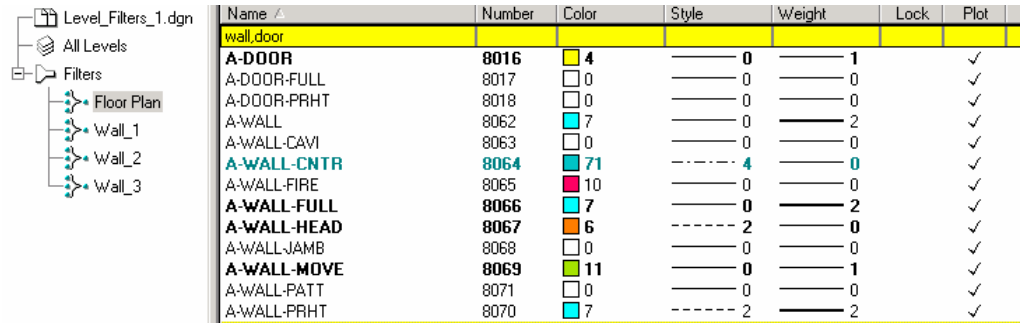
or….keyin: LV=# or….LV=name

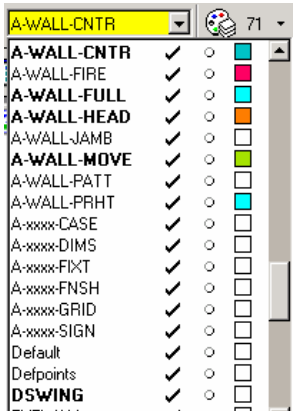
To manage most of the level features use the new LEVEL MANAGER dialog box.

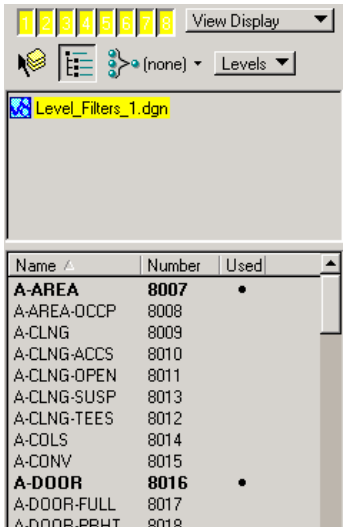
or…. keyin

OF=# or….
OF=name

ON=# or….
ON=name







AUTOCAD: Uses maximum of 32,000 layers. Layer Names can be 1-255 characters in length. These layers are typically named, but you could use numbers…..most AutoCAD users will use Layer Names….not Numbers.

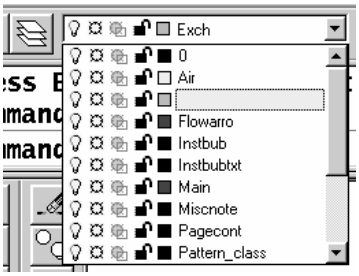
To change the CURRENT LAYER use the LAYER CONTROL on the Object Properties Toolbar:

or….keyin LAYER OFF LAYER ON

To turn levels ON or OFF use the following dialog box:

AutoCAD also includes a "layer 0" in every file. This layer is required and is used for special purposes, such as Blocks.

Another "different" layer found in AutoCAD is the DEFPOINTS layer. This layer is similar to "Construction Elements" in MicroStation. Anything found on the DEFPOINTS layer in AutoCAD will NOT PLOT regardless of the screen display. This layer is generated automatically when an associated dimension is placed. (It is the "definition point" or "snap point" on the object)



More LEVEL vs. LAYER

This one of the most important issues and most misunderstood areas between MicroStation and AutoCAD. While the need for levels in both MicroStation and AutoCAD is obvious, the use of layers in AutoCAD is CRITICAL!

AutoCAD relies on the layer structure to control data management, plot control, and WEIGHT (which didn't exist as an AutoCAD object property prior to AutoCAD 2000), COLOR, and LINETYPE. Users in AutoCAD differentiate the different properties between objects based primarily on layer. Hence, the meaning and purpose of the BYLAYER setting for both color and linetype.

Note: Some users in AutoCAD do draw objects on different layers while assigning specific colors and linetypes to individual objects. I call this the equivalent of drawing in MicroStation with 1 level, 1 color, and 1 line weight.

Yes, it is possible to do….but just because I can…should I?

LINESTYLE vs. LINETYPE

Here is another BIG DIFFERENCE! How each application handles different line styles is completely different.

MICROSTATION:

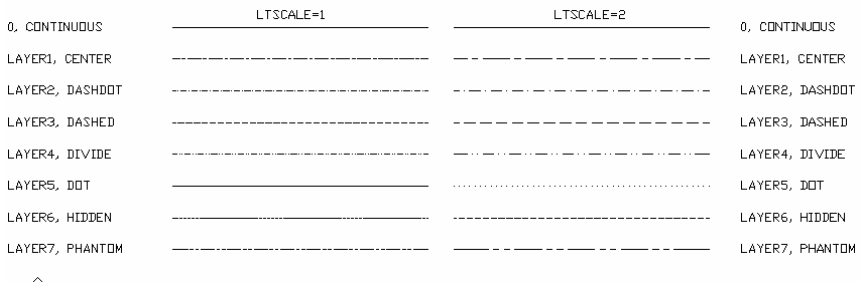
MicroStation may appear to have the fewest options when it comes to "out-of-the-box" line styles…or linecodes. But, you're wrong! With custom line styles I can easily create as many line styles as I want to. And the plus is how easy it is to do.

MicroStation also gives you an easy to use interface for creating custom line styles. However, be careful with how custom line styles translate! Basically….they don't, you have to drop them!

AUTOCAD:

AutoCAD has several delivered linetypes and a system variable called LTSCALE which allows the AutoCAD user to modify the "scale" of these linetypes globally throughout the drawing file.

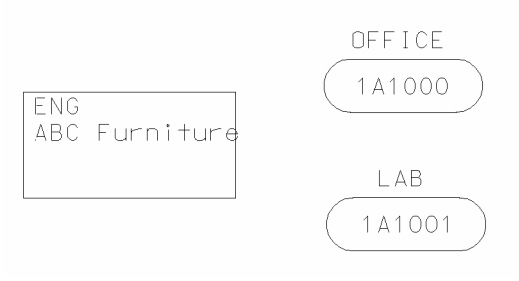
In R13, AutoCAD also introduced another setting called LINETYPE SCALE, which allows the user to modify the "scale" of a linetype on an object by object basis.

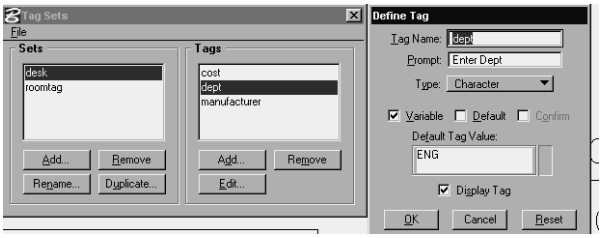


ENTER DATA FIELDS vs. ATTRIBUTE TAGS

This functionality is available on both sides of this discussion; they just call it something different.

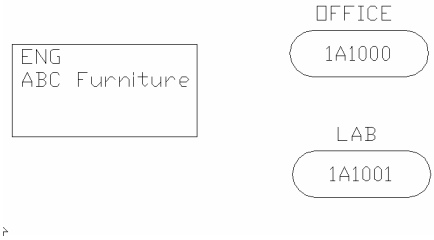
MICROSTATION: ENTER DATA FIELDS are generally used for editable text within cells. For example: an equipment tag, or room number. These cells have varying text data every time they are used, so it is critical that this be an easy edit to make without having to DROP the cell.

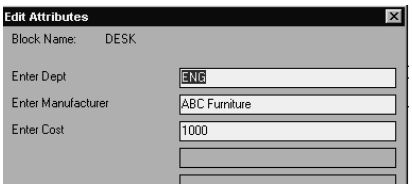




AUTOCAD: ATTRIBUTES are very similar to Enter Data Fields in MicroStation. They allow the AutoCAD user to edit the text inside of a block without having to EXPLODE the block.

Attributes can be edited globally, which is very convenient, however, DO NOT EXPLODE a block with attributes with the default AutoCAD EXPLODE command or the text data will be destroyed.

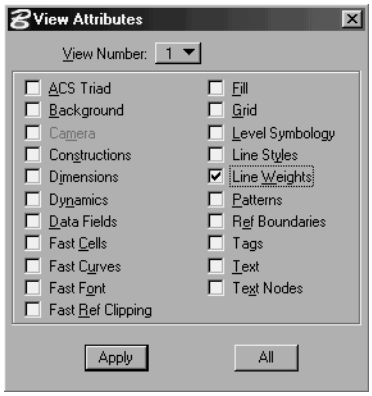




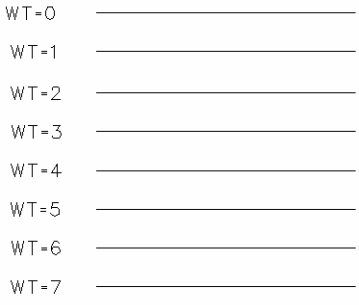
You must use an EXPRESS TOOL called “EXPLODE ATTRIBUTES TO TEXT” to maintain the text data during an explode.

Line WEIGHT vs. COLOR

MICROSTATION: Line weight has always existed in MicroStation as an element attribute. This allows you to see visual line weight on the screen while working. You can turn off screen line weight using the VIEW ATTRIBUTES command.



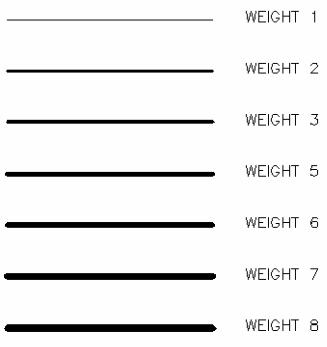
Line Weights - ON



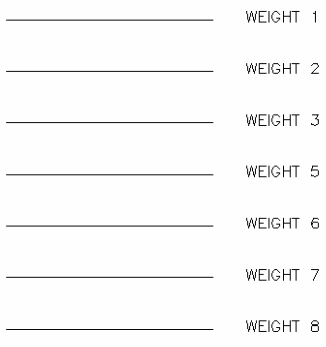
Line Weights - OFF

AUTOCAD: Screen line weight was never been available in AutoCAD (until 200x). To get line weight an AutoCAD user assigns "colors" to an object based on the layer, and then maps that color to a specific line weight during the PLOT process.

Screen line weight is now available in AutoCAD 200x. To plot line weight you can use this line weight assignment, or use the color to weight mapping method similar to previous releases of AutoCAD. Note that most AutoCAD users still don't utilize the line weight feature.



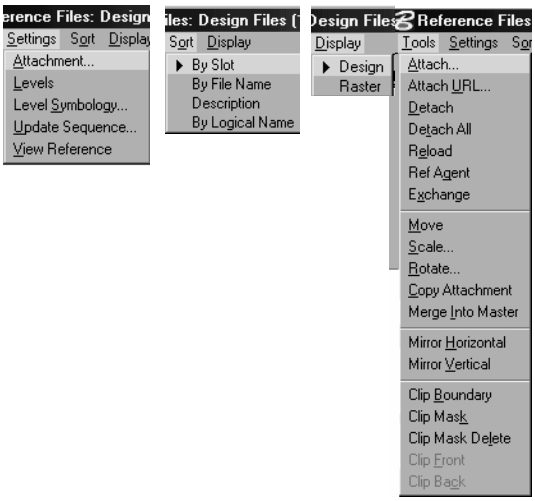
Line Weights - ON



Line Weights - OFF

Reference Files vs. Xrefs

Reference Files compare better than ever before! MicroStation still has more functionality and manipulations available than AutoCAD, but the gap is closing!



MICROSTATION: MicroStation supports the following reference file commands. The bolded items are unique to MicroStation.

ATTACH
DETACH
RELOAD
REF AGENT – maintains local copies of ref files attached from the web
EXCHANGE – opens the ref file for editing
MOVE
SCALE
ROTATE
COPY ATTACHMENT – copy the ref file attachment
MERGE TO MASTER – merge the ref file into the active or master file
MIRROR HORIZONTAL
MIRROR VERTICAL
CLIP BOUNDARY
CLIP MASK – hide a portion of the ref file
CLIP MASK DELETE – remove the clipping boundary from the ref file
NEST DEPTH – control how far down the file tree ref files are inherited
LOGICAL NAMES – provide a user-friendly name for various ref file tools
HILITE MODE – controls how ref files are visually identified in the graphics view

DISPLAY – control the display of the ref file
LOCATE – control the accessibility of elements in the ref file
SNAP – control snapping to elements in the ref file

AUTOCAD: Supports the following reference file commands. The bolded items are unique to AutoCAD.

ATTACH
DETACH
RELOAD
UNLOAD
MOVE
SCALE
ROTATE
MIRROR
CLIP
BIND
CLIPFRAME
XREF EDIT
COPY NESTED OBJECTS
TRIM TO XREF
EXTEND TO XREF
LIST XREF
EDIT XREFS PATHS
XREF MANAGER

Exercise: Learning the differences in Reference Files

This exercise will cover just some of the differences in how reference files are used. I think you will find that MicroStation has a much more powerful set of tools to work with reference files. You can do everything AutoCAD can do….and more! Let's take a look!

STEP 1: **File → Open**….**REFERENCE.DGN**

STEP 2: Select **File → Reference** and highlight the **LAND_WEST** reference file and <*Right-Click*> to explore the commands available

STEP 3: Select the **EXCHANGE** command to switch to the reference file for editing

STEP 4: To return to the "master" file…select **File → 2 – REFERENCE.DGN**…now you should be back in the "master" file

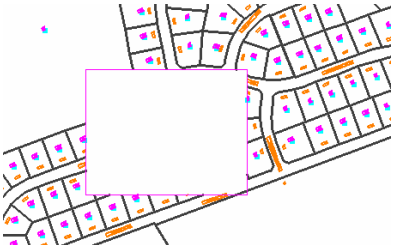
STEP 5: Select the **MERGE INTO MASTER** command and ***<Data-point>*** in a view to complete the merge

The view selected is more important than most users initially think. Only the levels that are displayed in that view are merged. All other levels are ignored.

STEP 6: Place a **FENCE** in the middle of the reference file

STEP 7: Select the **CLIP MASK** command and ***<Data-point>*** to accept the clip

You should see the portion of the reference file INSIDE the fence disappear.



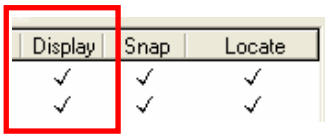
STEP 8: Select the **HIGHLIGHT MODE** and set it to **HIGHLIGHT**. This causes the selected reference file in the dialog to also highlight visibly in all views.

STEP 9: Select the **BOUNDARIES** mode and notice that only a boundary is highlighted in all views.

….or you can select **BOTH** and the reference files will highlight with a boundary in all views.

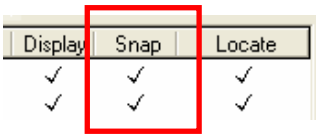
STEP 10: **Turn OFF** the display of **LAND_WEST** by selecting the ✓ in the **DISPLAY COLUMN** of the dialog box.

Select it again to **Turn ON** the display of **LAND_WEST**.



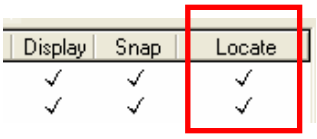
STEP 11: **Turn OFF** the ability to **SNAP** to elements in the reference file by selecting the ✓ in the **SNAP COLUMN** of the dialog box.

Select it again to **Turn ON** this feature for **LAND_WEST**.



STEP 12: **Turn OFF** the ability to **COPY** elements from the reference file by selecting the ✓ in the **LOCATE COLUMN** of the dialog box.

Select it again to **Turn ON** this feature for **LAND_WEST**.



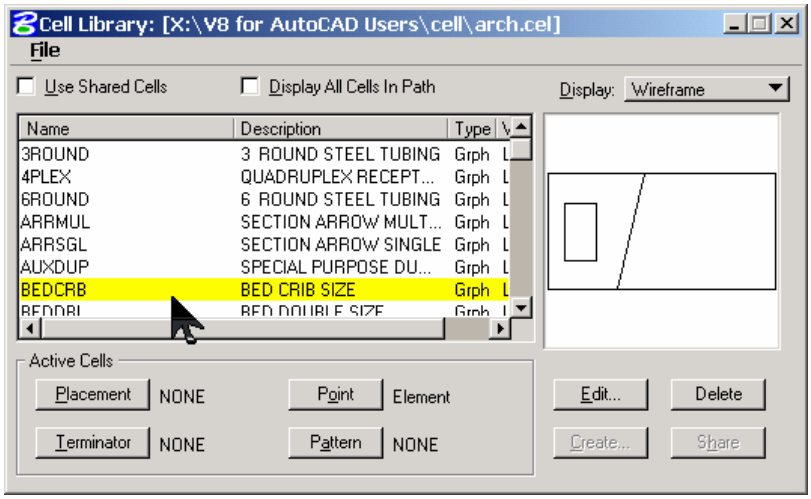
Cell Libraries – DesignCenter

The storage of standard symbols is critical to either application. Both provide more than one way to retrieve and populate standard symbol libraries.

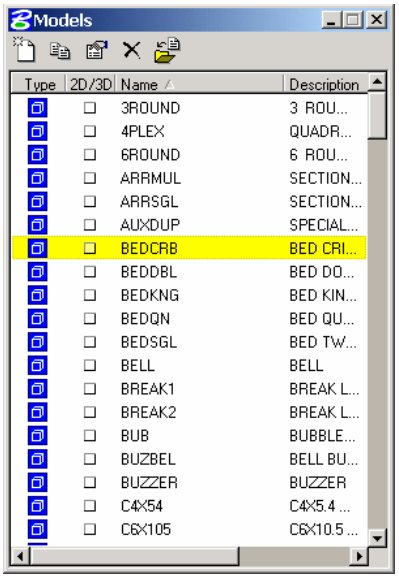
MICROSTATION: A cell library is the primary method for storing symbols in MicroStation. However, with the release of V8, you can now store symbols in models as well.

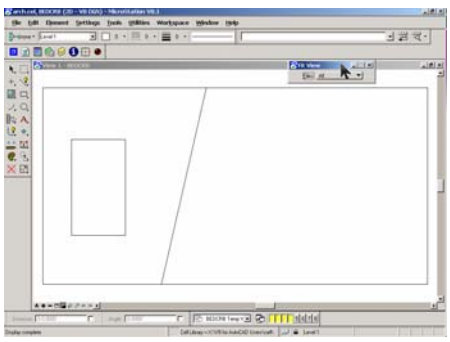
Cell libraries as we have known them in the past are really gone. But the cool thing about the transition is that Bentley has made this change virtually transparent to the user. You may think that you are using the same old cell library...but you're not! What you see in the cell library dialog can be located from various locations.

The cell library file format no longer exists. The .CEL file format is really the same as a .DGN file. One new feature allows the user to EDIT cells while in the "library".



Here is what the file looks like if you look at it from a new perspective. You can edit the cell and the "library" updates automatically. This new concept might take you a few minutes to master, but you can do it!





You can also use the MicroStation tool CELL SELECTOR which is very similar to the DESIGN CENTER…only BETTER!

The CELL SELECTOR Tool allows you to create a "tool palette" from cells so that you can visually pick cells easily for placement. CELL SELECTOR also allows you to control the settings and properties automatically during placement.

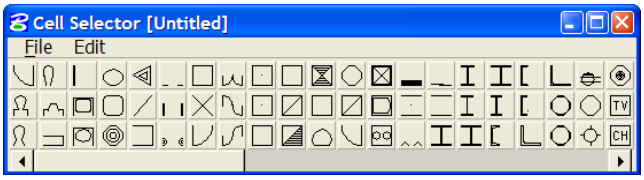
Exercise: Using CELL SELECTOR

This exercise will demonstrate how to set up CELL SELECTOR to enhance cell placement in MicroStation.

STEP 1: **File → Open**….**CELLS.DGN**

STEP 2: Select **Utilities → Cell Selector**

STEP 3: Select any cell from the created dialog box and place it in the design file.



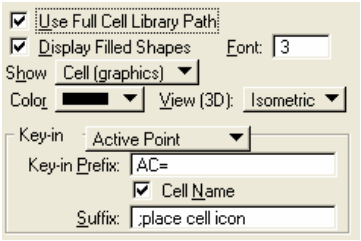
STEP 4: From the CELL SELECTOR dialog box…select **Edit → Button Size**

STEP 5: Modify the **BUTTON SIZE** to **24** and the **GAP SIZE** to **2**

STEP 6: Pick **OK** to save the changes

STEP 7: From the CELL SELECTOR dialog box…select **Edit → Defaults**

You can modify several settings here to manage and control the cell placement. All you CAD Managers should really like this one!

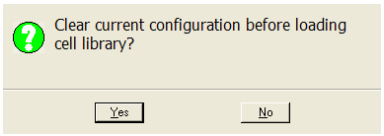


You can pick and choose which cells you want in the CELL SELECTOR dialog box, as well as import cells from several cell libraries into a single CELL SELECTOR dialog box.

STEP 1: Select **File → Load Cell Library**

STEP 2: Select **AREAPAT.CEL** from the list and pick **OK** to complete

STEP 3: Select **NO** to ADD the new cells to the current CELL SELECTOR dialog box



STEP 4: To REMOVE cells from the **CELL SELECTOR** dialog, select the cell button and press the ***<Delete>*** key on the keyboard

After setting up the CELL SELECTOR for your company's use, you can SAVE the settings for everyone to use.

STEP 1: From the CELL SELECTOR dialog box…select **File → Save As**…and save your CELL SELECTOR preferences to an external file

Standard Cells.CSF

This external file can be placed on a server so that it can be easily accessed by everyone in your organization. The following configuration variable will load the CELL SELECTOR file automatically for all users.

STEP 1: Select **Workspace → Configuration**

STEP 2: Select the category **CELLS** and highlight the setting for **Cell Selector File**

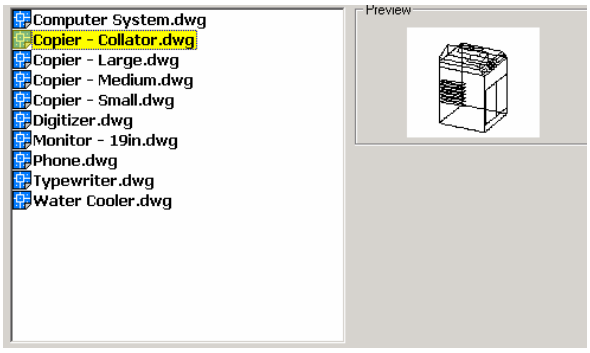
STEP 3: Pick the **SELECT** button and navigate to your external CELL SELECTOR file

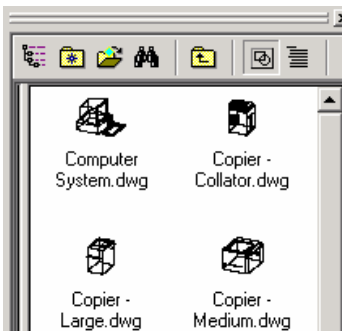
STEP 4: Pick **OK** and **YES** to save your changes

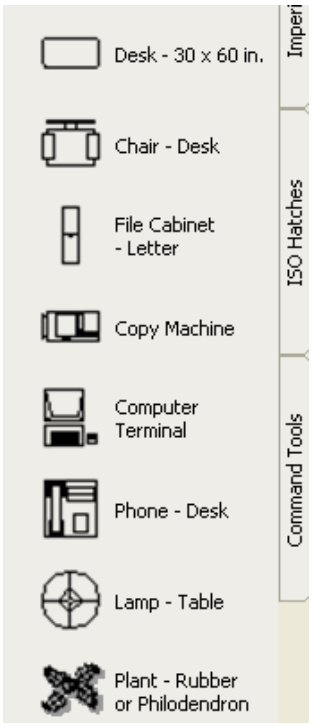
AUTOCAD: AutoCAD also provides three primary methods for inserting standard symbols.
WBLOCKS Design Center Tool Palettes

First , you can INSERT blocks from WBLOCK files; where each block is stored in a separate DWG file….. or from DWG files containing several blocks using the new DESIGN CENTER.

Here is a sample of each:







What's NEW….

MicroStation V8 2004 vs. AutoCAD 2004/2005

DGN/DWG Enhancements

MicroStation V8 2004 will read and write to 2004 file formatted files. The following features are now supported between CAD files.

- Associative Dimensions
- Layer Filters
- Multi-line styles
- Wipeout Objects
- Password protected DWG files
- DWG file recovery
- DWG Audit

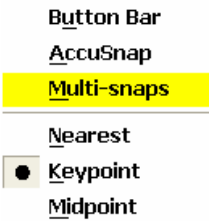
Misc V8 2004 Features/Commands

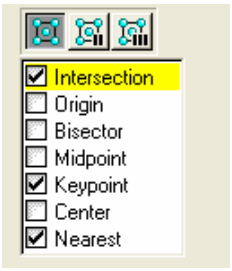
Some of the following commands have also been enhanced with MicroStation V8 2004.

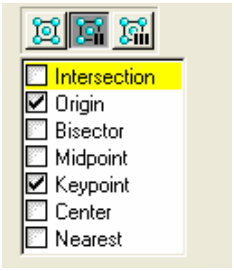
- Ability to read and write DWS (drawing settings) files
- Ability to control the Seed File Global Origin
- Ortho/Axis compatibility
- Reference File Logical Names mapping
- True Line weight mapping
- Convert Empty Data Fields to Spaces
- UCS compatibility
- Front and Back Clipping Planes
- Text Style names
- Convert Reference Files

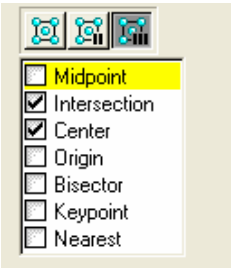
Multiple SNAPs vs. OSNAPs

MICROSTATION: MicroStation V8 2004 introduces multiple snap capabilities similar to those found in AutoCAD, only better. The ability to configure 3 "saved" snap combinations to provide the user with simultaneous snap conditions is a very convenient and productive application for the user.

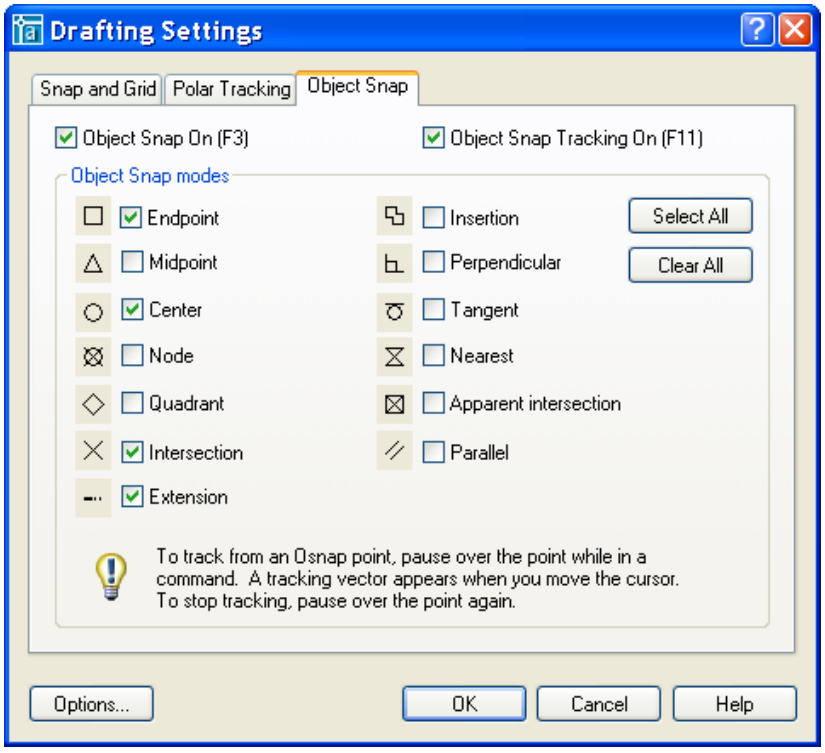






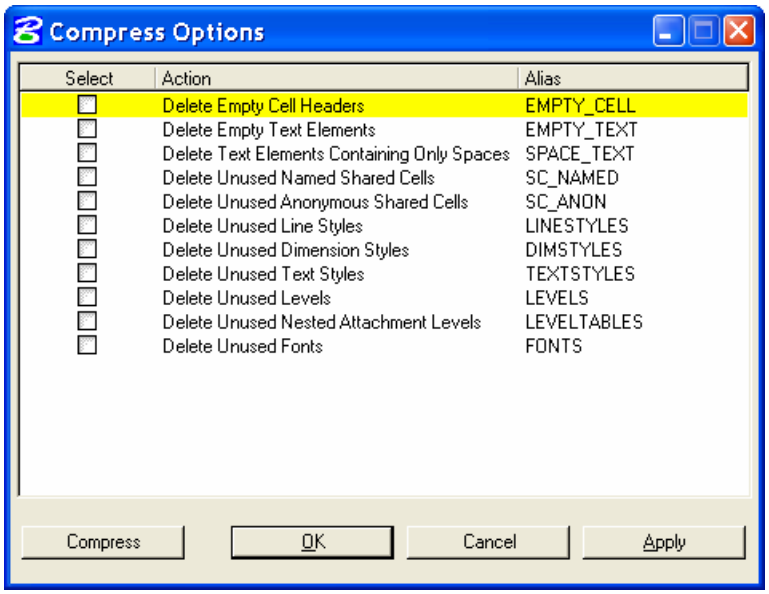


AUTOCAD: AutoCAD provides the ability to have simultaneous osnaps set at one time as shown below.

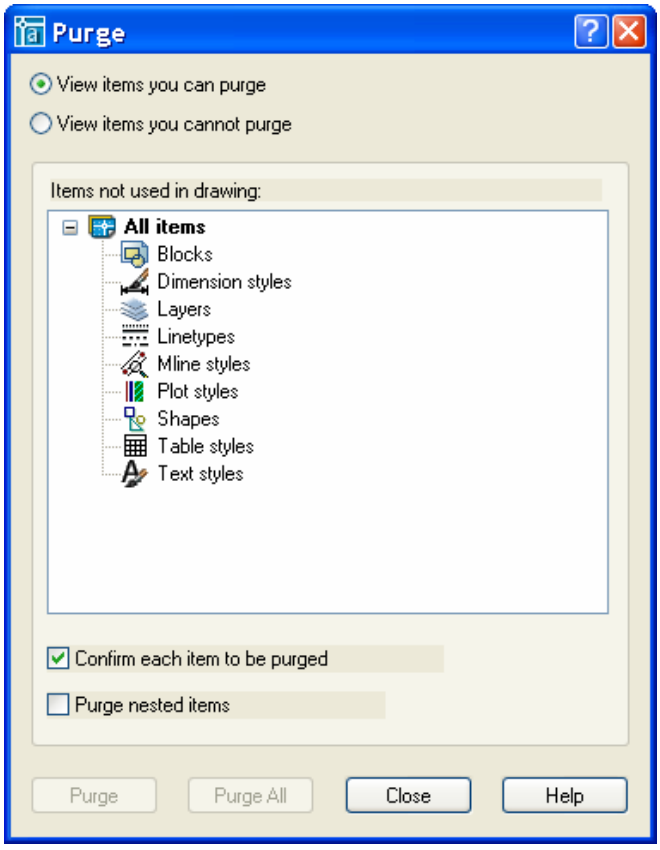


DGN Compress vs. Purge

MICROSTATION: MicroStation V8 2004 also provides new file compression options similar to the AutoCAD Purge command.

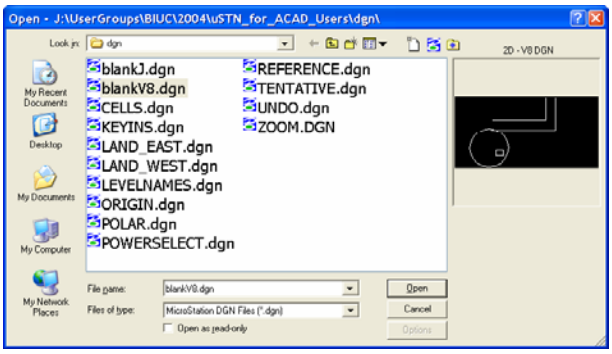


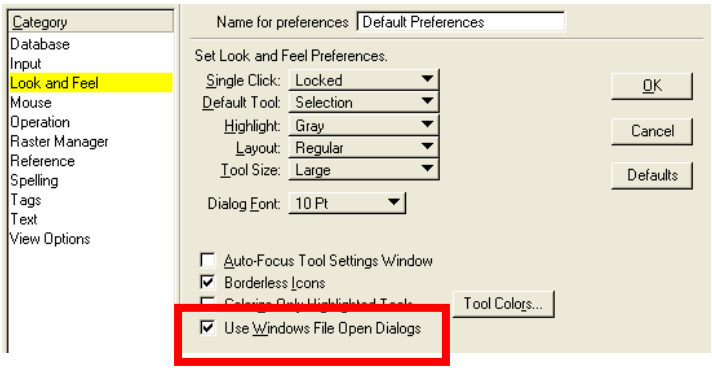
AUTOCAD: AutoCAD allows for the user to select which aspects of the file to purge and remove.

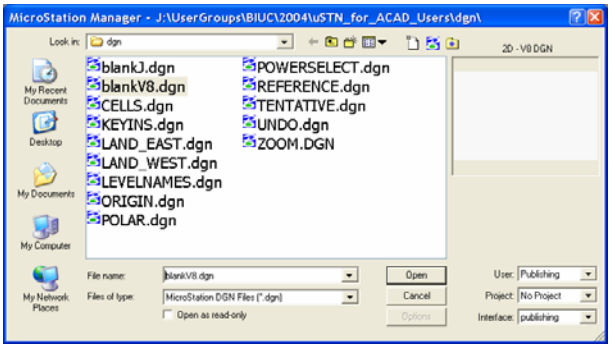


Windows-Like Interface in BOTH

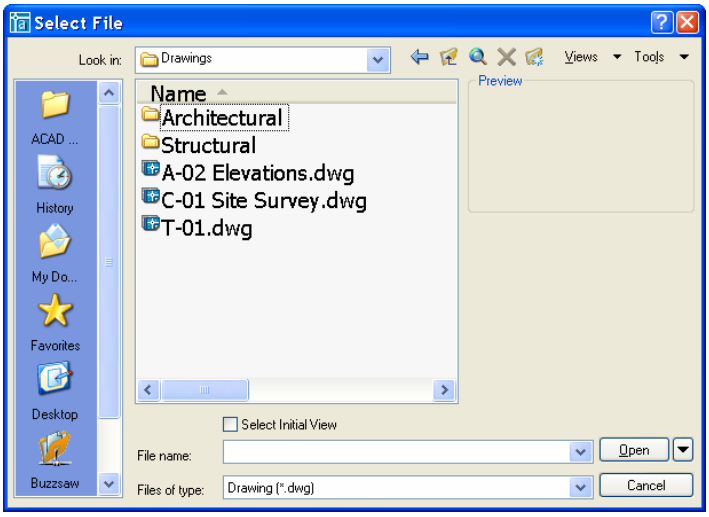
MICROSTATION: The file operation dialogs can be modified to be more “windows-like” via a new preferences in V8 2004. Check out the following preference settings to modify the default interface.





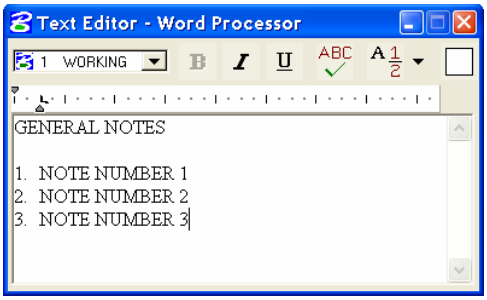


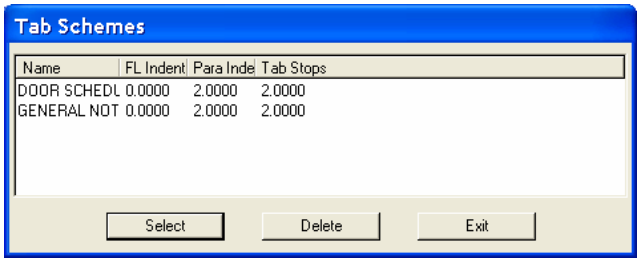
AUTOCAD: The file dialogs in AutoCAD are “windows-like” by default, and are not customizable.



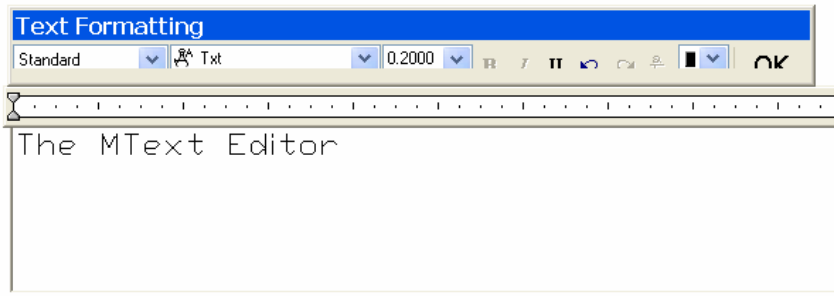
Text Editor vs. MText Editor

MICROSTATION: The new Text Editor in V8 2004 is similar to the MText Editor found in AutoCAD in respect to Tabs and Indents. However, in typically fashion, they took it a step further with the availability of Tab Schemes.





AUTOCAD: The MText Editor available in AutoCAD

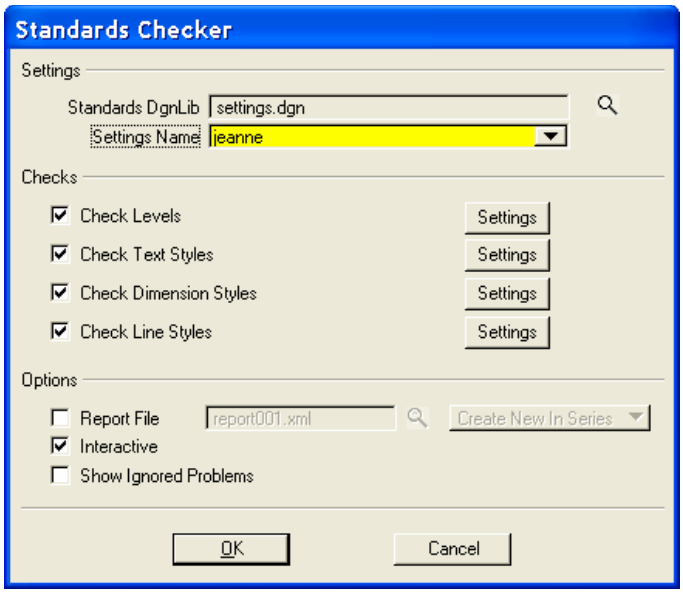


<Double-Click> Editing

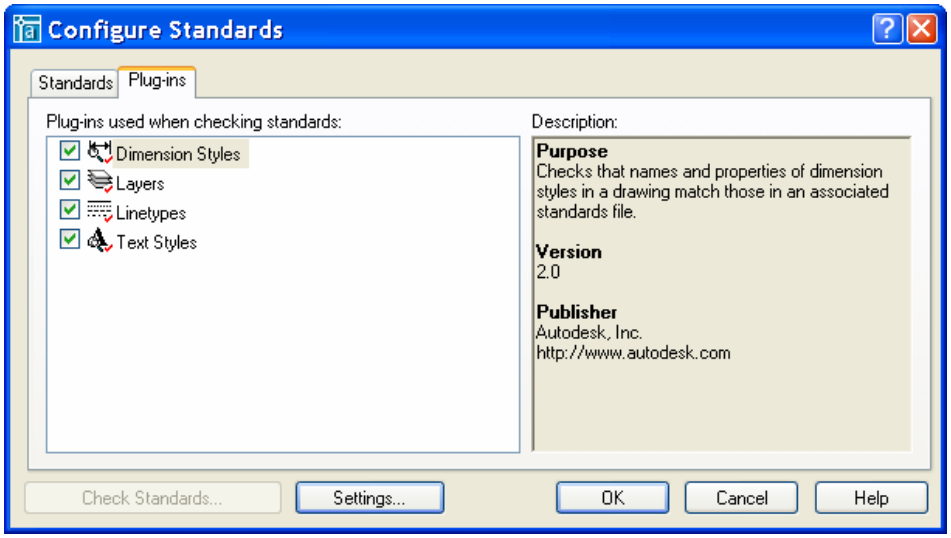
You will be able to *<Double-Click>* on **text elements** to edit them in V8 2004.

Standards Checker in BOTH

MICROSTATION: V8 2004 provides the ability to check drawings for standards compliance.

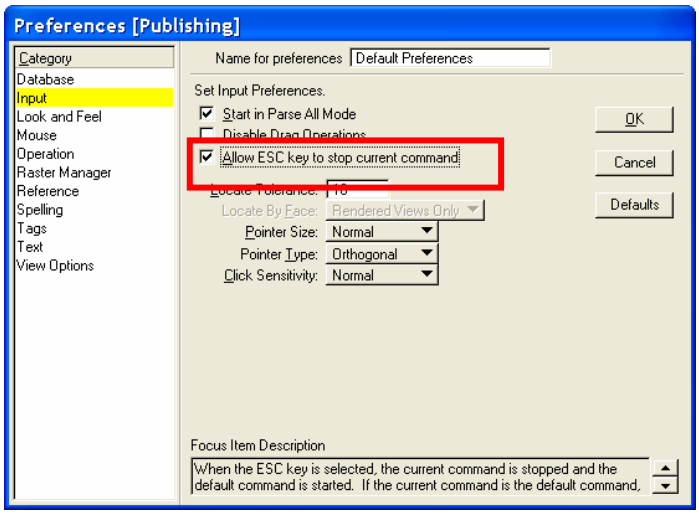


AUTOCAD: The CAD Standards Checker available in AutoCAD.



ESC to Cancel in BOTH

MicroStation V8 2004 provides a user preference to allow the AutoCAD user to set the ESC key to cancel any command in MicroStation.



Question & Answers

If time allows….at this point I will open it up for general questions and answers.

As you can see MicroStation has most of the same functionality that you had in AutoCAD…plus more! I hope I was able to show you some tools that will help you get comfortable quicker and with the least amount of "pain". Remember, most of our frustration is "emotional", and the sooner you take the plunge the better. Have fun…I feel confident that you will be extremely happy with what MicroStation has to offer.

WELL…..that's probably more than I have time for….but hopefully not more that you wanted to know! I hope this has been a benefit to each of you.

Thank you for your time

Hope you enjoyed the session!

For high quality and affordable Training and Consulting Services

Contact:

Jeanne Aarhus

ProSoft NET - Omaha
12005 Quail Drive
Omaha, Nebraska, 68123-1175

402-292-6016

jaarhus@prosoftnet.com